

FreeCAD-CFD Workbench

Tutorial 3: Boundary layer meshing and turbulent flow



CFD Workbench

WORKBENCH:

This workbench aims to help users set up and run CFD analysis. It guides the user in selecting the relevant physics, specifying the material properties, generating a mesh, assigning boundary conditions and setting the solver settings before running the simulation. Where possible best practices are included to improve the stability of the solvers.

PREREQUISITES:

WINDOWS:

- Install the binary. All necessary software components are included.
- Install blueCFD build of OpenFOAM (<http://bluecfd.github.io/Core/Downloads>)

LINUX:

- FreeCAD (https://www.freecadweb.org/wiki/Install_on_Unix)
- OpenFOAM (4.1.0 or later) (<https://openfoam.org/download/>)
- Paraview (tested with 5.0.1)
- Gnuplot (tested with 5.0)
- GMSH (2.13+)
- cfMesh (1.1.2)

For more information, view the CFD workbench README file.

DEVELOPERS:

Johan Heyns (CSIR, 2016) jheyns@csir.co.za,

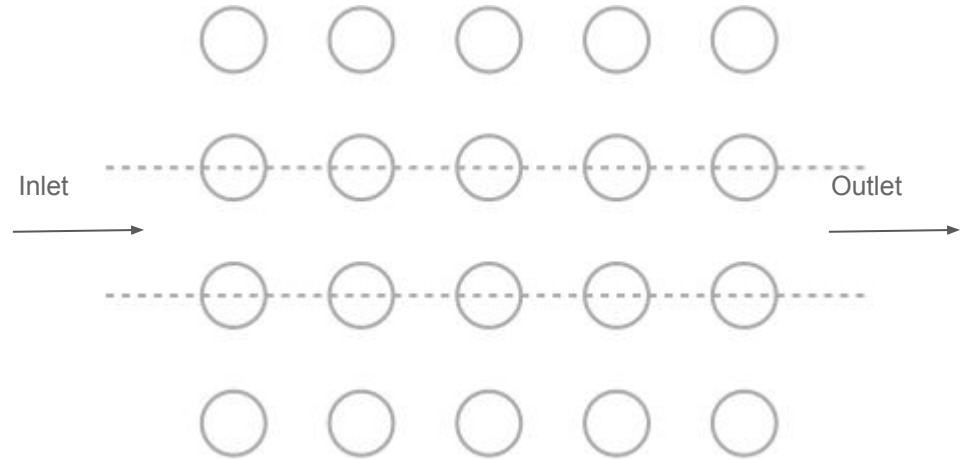
Oliver Oxtoby (CSIR, 2016) ooxtoby@csir.co.za,

Alfred Bogaers (CSIR, 2016) abogaers@csir.co.za,

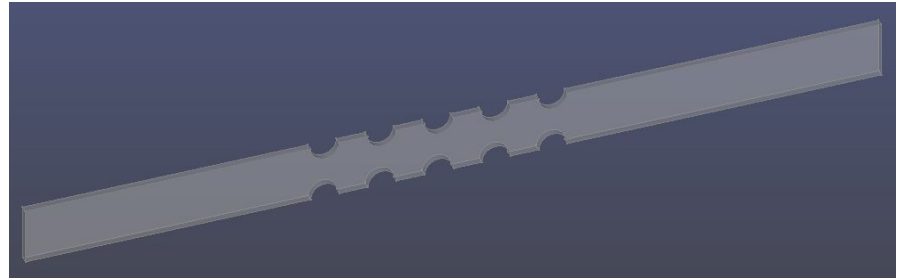
Part Design

Tube bundle

- To demonstrate how to model viscous flow, the pressure drop over a tube bundle is calculated.
- Bundle parameters
 - Tube diameter: 100 mm
 - Tube pitch: 200 mm
- Assuming symmetric conditions we model the flow through one of the channels.



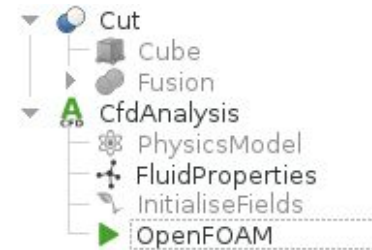
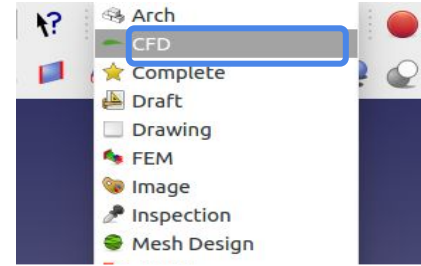
- Activate the “Part” workbench
- Create a cube
 - Length: 3000 mm
 - Width: 200 mm
 - Height: 25 mm
- Create 10 cylinders
 - Diameter: 100 mm
 - Spacing: 200 mm
- For the channel cut the cylinders away from the cube



Boundary layer meshing

Initialise CFD analysis

- Activate the CFD WB by clicking on the dropdown menu in the taskbar and select “CFD”
- Create a CFD analysis which will automatically generate:
 - Physics model
 - Fluid properties
 - Initialise fields
 - OpenFOAM solver

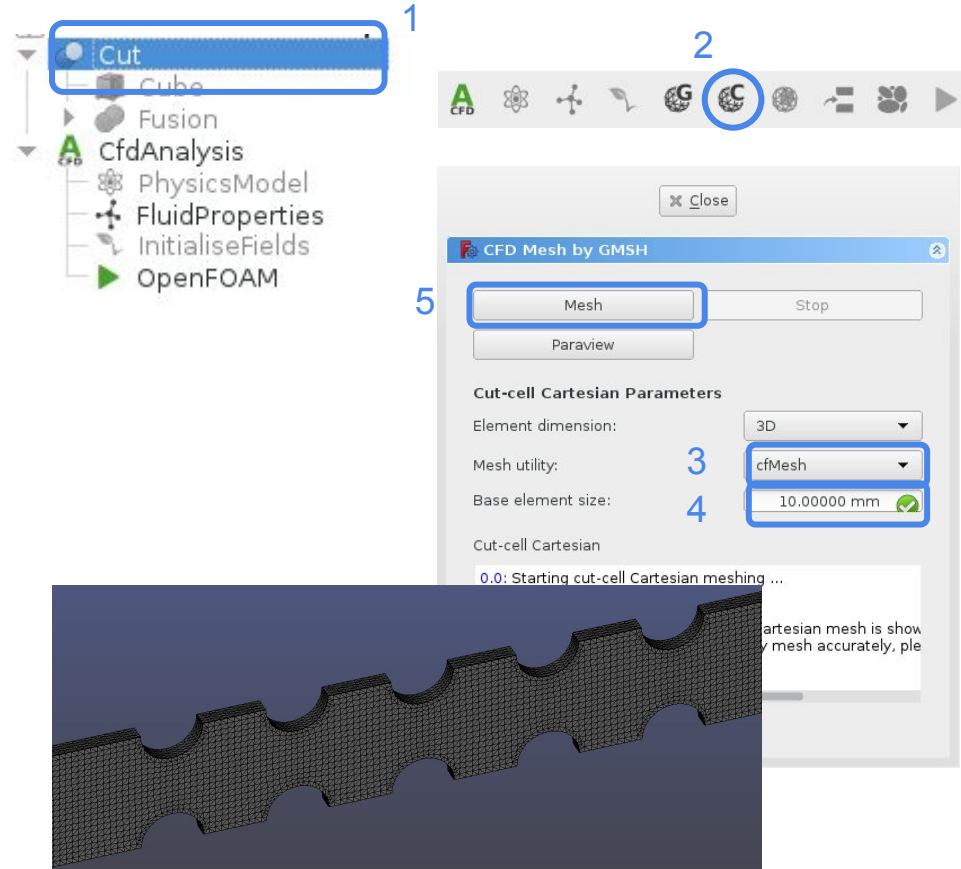


Create an initial mesh


- To create a preliminary mesh select the 'Cut' object.
- Click on the cut-cell Cartesian icon
- Select the 'cfMesh' utility which allows for boundary layer meshing.
- Set the maximum element characteristic length to 10 mm and mesh.
- You should now have a uniformly spaced cut-cell Cartesian mesh.

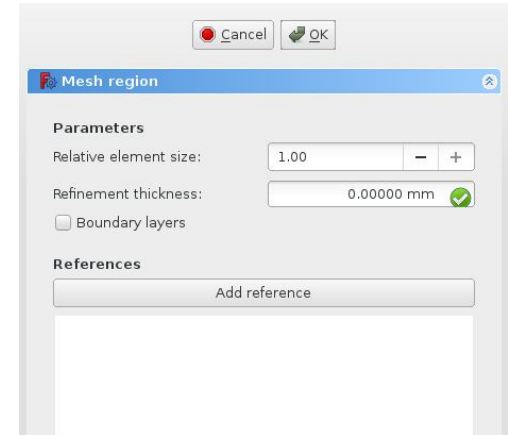
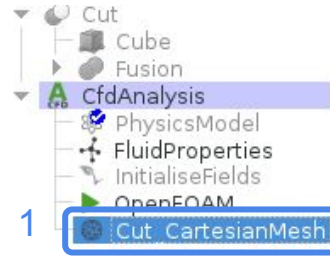
NOTE: Currently, only the 'cfMesh' utility supports boundary layer meshing.

NOTE: Currently, Cartesian meshing only supports 3D meshing



Adding boundary layers

- Highlight the mesh object to activate the mesh region icon 
- Create a mesh region by clicking on the icon .
- A mesh region object will now have been created (if not visible expand “Cut_CartesianMesh”).
- Double click it to open the task panel.
- In this task panel, we can now edit the refinement and boundary layer parameters.



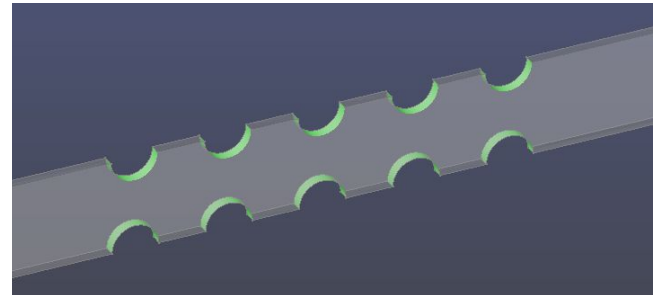
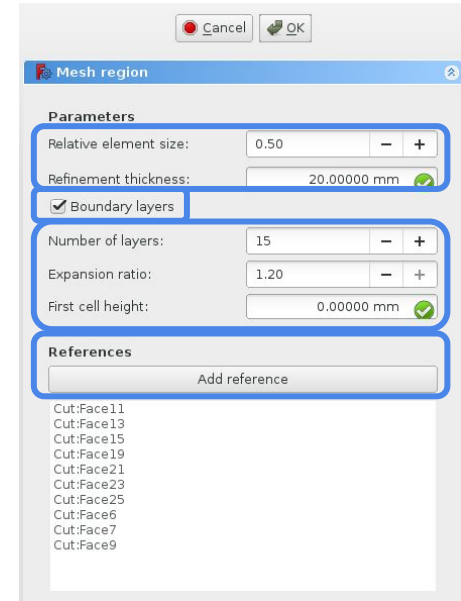
Mesh region parameters

- The refinement parameters ‘Relative element size’ and ‘Thickness’ are respectively used to set the cell size relative to the base mesh and the absolute thickness of the refinement region.
 - Relative element size: 0.5
 - Refinement thickness: 20 mm
- Check the “Boundary Layer” box
- The user can set the number of layers, the expansion ratio that governs the growth of the layers and, optionally, the maximum first cell height (The default 0 mm will be ignored).
 - Number of layers: 15
 - Expansion ratio: 1.2
- Next, the reference faces need to be added.

NOTE: Currently, ‘cfMesh’ only supports surface refinement.

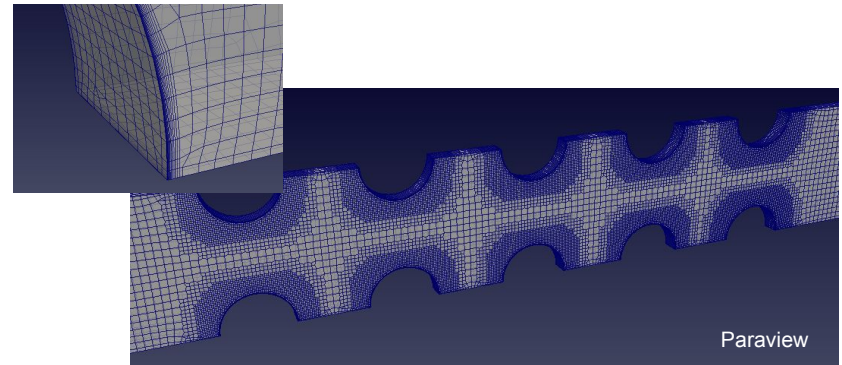
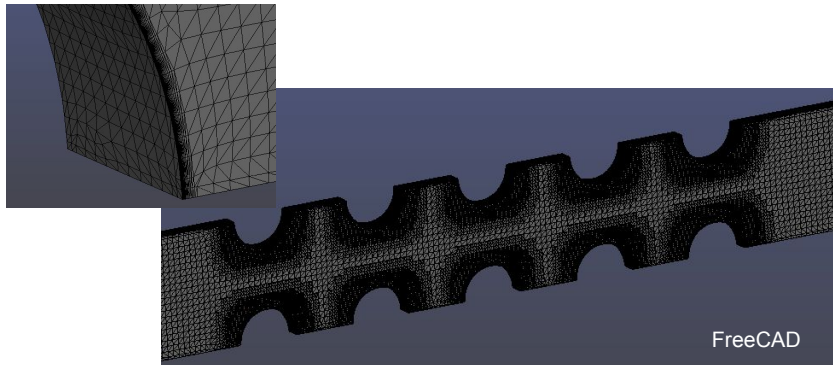
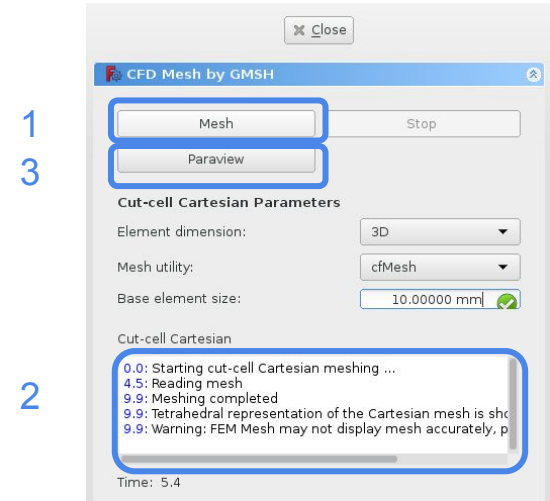
NOTE: The expansion ratio is limited to be greater than 1.0 and less than 1.2.

1
2
3
4



Update mesh

- Once the mesh region parameters are updated the user can go back to the mesh task panel and click on 'Mesh' to recompute.
- The message console shows the progress and once meshing is completed a tetrahedral representation of the mesh is displayed.
- For a more accurate representation of the mesh click on "Paraview" to open the viewer.



Viscous analysis

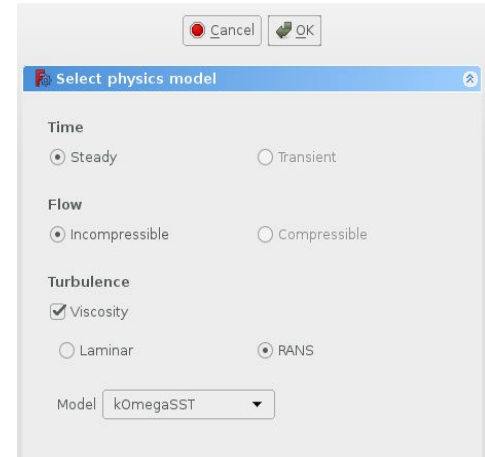
RANS analysis

- In the “PhysicsModel” task panel under “Turbulence”, select the RANS checkbox which will automatically select the k-w SST model.

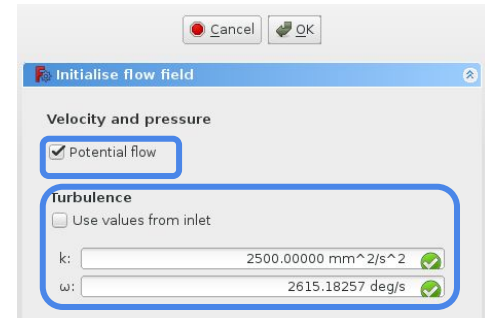
NOTE: Currently the CFD WB only supports the k-w SST model.

- Activate the “InitialiseFields” task panel and ensure the “Potential flow” checkbox is selected. The high aspect ratio boundary layer cells require a well initialised velocity field.
- The k and w can be initialised using the values from the inlet or specified by the user.

1



2



3

Boundary conditions

- In addition to entering the velocity at the inlet boundary, the user is required to specify the turbulence quantities.
- The user can either specify the Turbulence intensity and Length scale or the Kinetic energy (k) and Dissipation rate (w).

NOTE: The following links might be useful in guiding the user in specifying the inlet turbulent quantities:

<https://turbmodels.larc.nasa.gov/sst.html>

https://www.cfd-online.com/Wiki/Turbulence_intensity

https://www.cfd-online.com/Wiki/Turbulence_length_scale

- For the test problem, we will use turbulent intensity
 - Turbulent intensity: 5 %
 - Length scale: 2 mm



Turbulence specification

Intensity & Length Scale ▼

turbulence intensity and eddy length scale

Turbulence intensity (I) 0.05000 ✓

Length scale (l) 2.00000 mm ✓

Turbulence specification

Kinetic Energy & Specific Dissipation Rate ▼

k and omega specified

Turbulent kinetic energy (k) 1080.00000 mm²/s² ✓

Specific dissipation rate (ω) 57.29578 deg/s ✓

Boundary conditions

- Viscous/no-slip boundaries should be applied to the tube walls.
- For no-slip walls, wall functions are used for the turbulent quantities. This requires the y^+ of the first cell to be between 30 and 150.

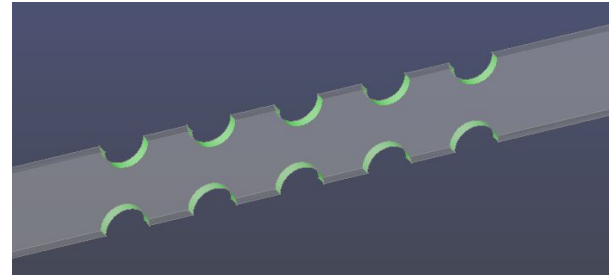
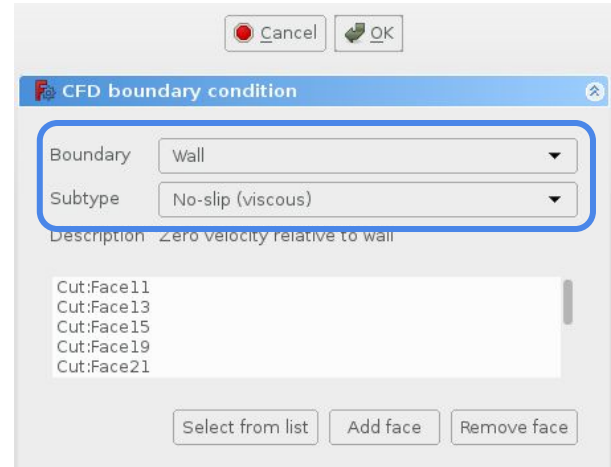
NOTE: The following links might be useful in guiding the user to compute the non-dimensional y^+ value:

[https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_\(y_plus\)](https://www.cfd-online.com/Wiki/Dimensionless_wall_distance_(y_plus))

https://www.cfd-online.com/Wiki/Law_of_the_wall

- The remainder of the walls may be treated as either “Slip” or “Symmetric” and “Static pressure” is prescribed at the outlet.

NOTE: An undefined boundary patch will default to slip/inviscid wall.



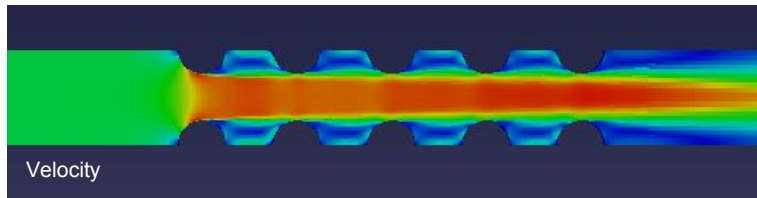
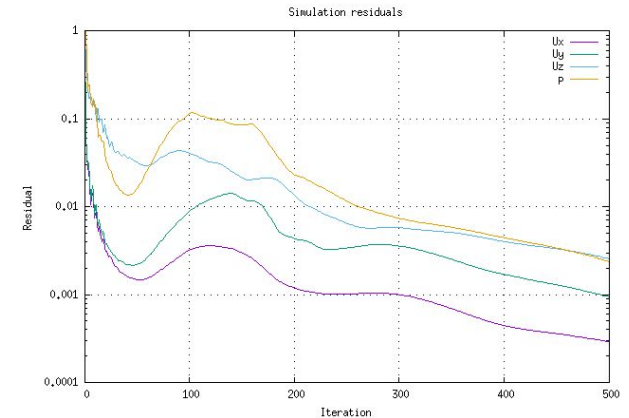
Running the simulation

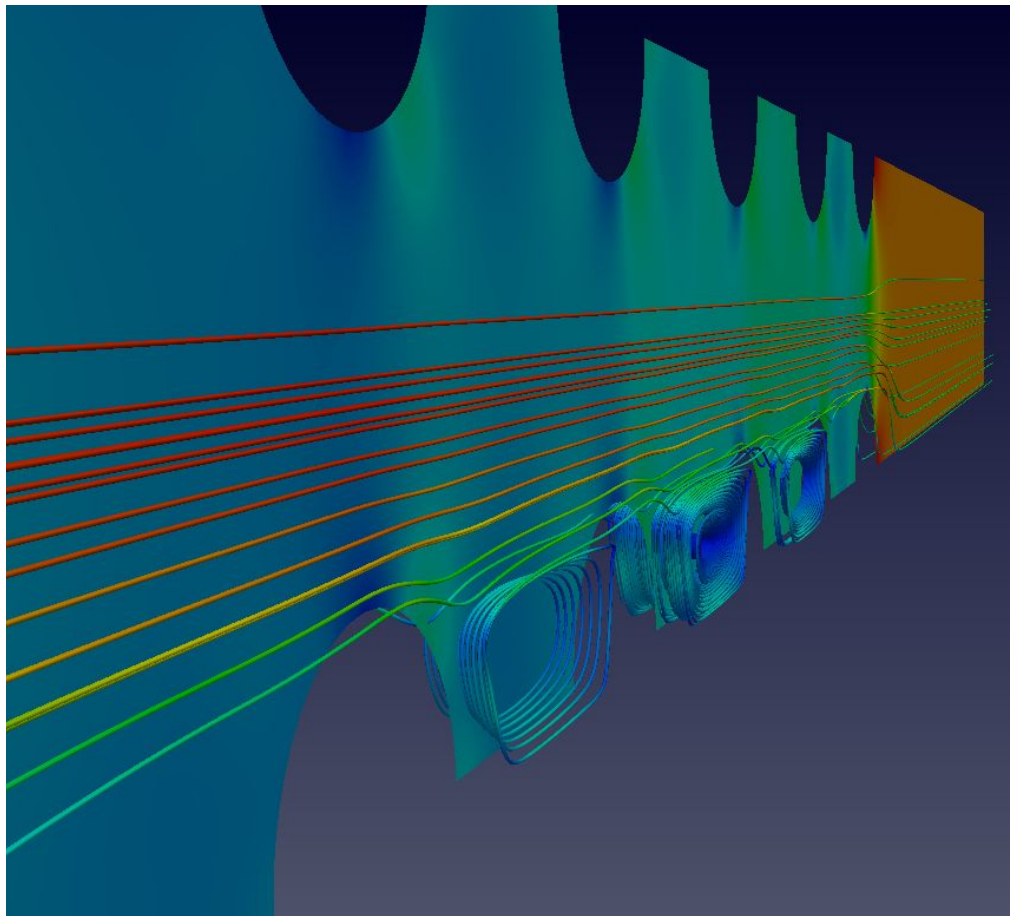
Simulation

- In the “OpenFOAM” data panel, the parallel setting can be set to true to run over multiple cores.
- The solver Task panel may then be activated, from which the case is written and simulation run.
- The residual plot is automatically created and the results can be viewed in Paraview.

NOTE: The incompressible solver in OpenFOAM stores the kinematic pressure, $p = P/\text{density}$.

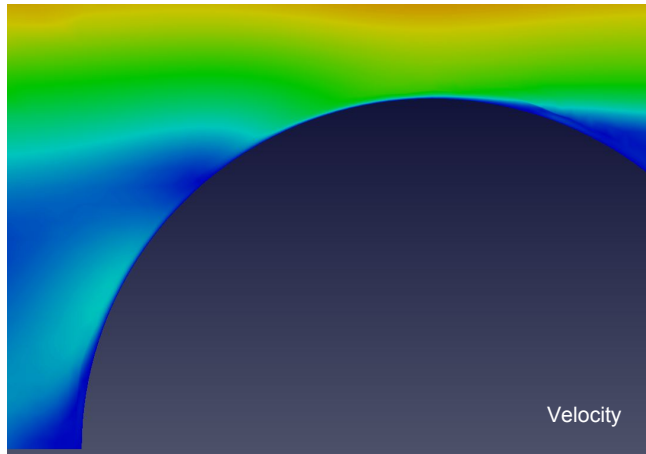
Solver	
Input Cas...	case
Parallel	true
Parallel C...	4
Solver N...	OpenFOAM
Working Dir	/tmp
Time Ste...	
Converg...	0.00010
End Time	500.00000
Time Step	1.00000
Write Inte...	10.00000





Paraview:

- “Slice” to show the pressure contours.
- “StreamTracer” to show the streamlines
- “Tube” to apply thickness to the streamlines



The End